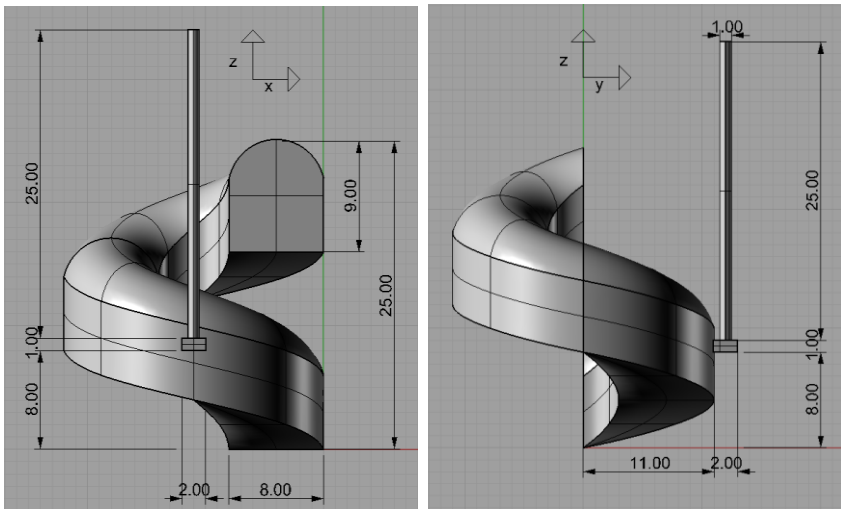
16. Tutorial 11 - Importing a Model from *Rhinoceros 4.0*



Rhinoceros 4.0 (*Rhino*) is a NURBS based solid modelling package for Windows that can create 3D shapes and surfaces with no limit on complexity.

In this tutorial, the above model will be created in *Rhino* and imported into *Examine*^{3D}.

If you have not already done so, start up *Rhino* by selecting the Rhinoceros 4.0 icon in the Start → Rhinoceros 4.0 menu.

Select **Large Objects – Metres** from the Startup Template.

Model

This model represents a horseshoe shaped spiral tunnel 9 metres tall and 8 metres wide. The tunnel is connected to a 20 metre cylindrical ventilation shaft 0.5 metres in radius by a 2 metre by1 metre by 2 metre rectangular vent that shall be referred to as a "rectangular elbow."

Project Settings

The default settings in *Rhino* are not suitable for generating quality meshes for *Examine*^{3D}. Change the project settings by selecting File → Properties. Select **Units** from the left pane of the window. Set **Absolute tolerance** to 0.0001 units, **Relative tolerance** to 0.001 percent, and **Angle tolerance** to 0.001 degrees. Hit OK to exit the properties window.

In the bottom-left hand corner of the screen, click on "CPlane" to turn on the "World" coordinate system.

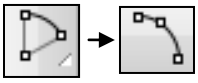
Creating the Tunnel

Let's start our design by establishing the shape of our tunnel, as follows:

Select: Curve → Polyline → Polyline

Enter the following in the prompt line at the top of the *Rhino* workspace.

```
Start of polyline: 0,0,6
Next point of polyline (Mode=Line Helpers=No Undo ): 0,0,0
Next point of polyline. Press Enter when done (…): 8,0,0
Next point of polyline. Press Enter when done (…): 8,0,6
Next point of polyline. Press Enter when done (…): Enter
```

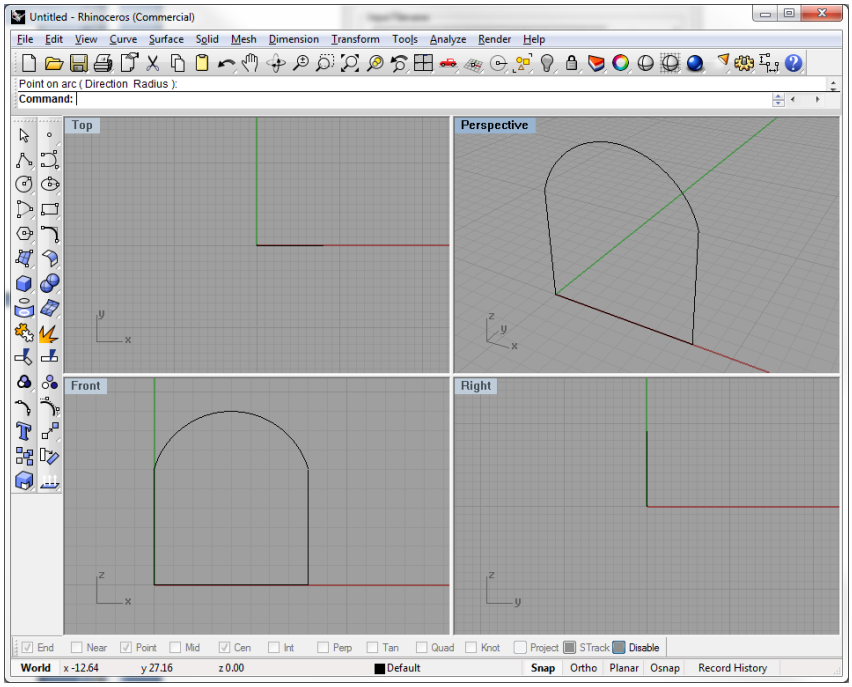


Select: Curve → Arc → Start, End, Point

Enter the following in the prompt line at the top of the *Rhino* workspace.

```
Start of arc: 0,0,6
End of arc: 8,0,6
Radius and orientation of arc <default>: 4,0,9
```

You should now see the shape of the tunnel clearly in the "Front" and "Perspective" view panes.


Untitled - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Point on arc (Direction Radius):
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x -12.64 y 27.16 z 0.00 Default Snap Ortho Planar Osnap Record History

Join

Although it looks like we've built a solid tunnel cross section, the arc and polyline are not connected. The **Join** command must be used to "glue" the two curves together into one shape.

Select Edit →Join

Click on the arc and the polyline. We have now created a unique shape from two simple curves.

Copy, Paste, Move, and Rotate

We shall now create the "framework" for our curved tunnel by moving and rotating copies of our original tunnel shape. Activate the "top" view pane by clicking anywhere in its bounds. Ensure the tunnel cross section is selected.

Select: Edit → Copy

Select: Edit → Paste

Select: Transform → Rotate

Enter the following in the command window.

```
Centre of rotation (Copy): 4,0,0
Angle of first reference point (Copy): 8,0,0
Second reference point (Copy): 4,-4,0
```

We now have a copy of our original tunnel shape rotated 90 degrees. Let's move it to define the midpoint of the tunnel's curve.

Select: Transform → Move

Enter the following in the command window.

```
Point to move from (Vertical=No): 4,-4, 0
Point to move to <default>:  11,3,4
```

Note that we have raised this cross section 4 metres, indicating that our tunnel will be curving upwards. To make sure we get a smooth turn in our tunnel, we have to define another cross section at a 45 degree angle between the two cross sections we have defined. Select the line on the x-axis representing the original tunnel cross section.

Select: Edit → Copy

Select: Edit → Paste

Select: Transform → Rotate

Enter the following in the command window.

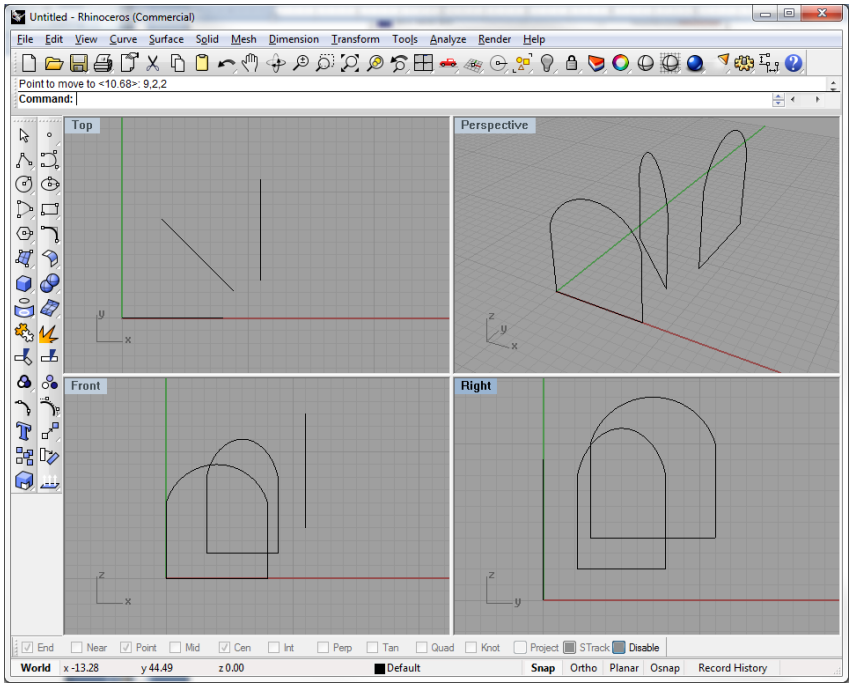
```
Centre of rotation (Copy): 4,0,0
Angle of first reference point (Copy): 8,0,0
Second reference point (Copy): 8,-4, 0
```

Select: Transform → Move

Enter the following in the command window.

```
Point to move from (Vertical=No): 7,-3, 0
Point to move to <default>:  9,2,2
```

Your cross sections should appear as shown below.


Untitled - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Point to move to <10.68>: 9,2,2
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x -13.28 y 44.49 z 0.00 Default Snap Ortho Planar Osnap Record History

Mirror

Instead of defining the other half of the tunnel using copy, paste, and rotate, let's use the mirror command to quickly reflect our cross sections and define the other half of our curve.

Select: Transform → Mirror

Select the horizontal and 45 degree cross sections in the "Top" view pane. Enter the following in the command window.

```
Start of mirror plane (3Point Copy=Yes): 11,12,0
End of mirror plane (Copy=Yes): 11,11,0
```

We now have a symmetrical series of cross sections! The 2 new cross sections must be adjusted, however, to continue the upward spiral of the tunnel. Select the new 45 degree cross section.

Select: Transform → Move

Enter the following in the command window.

```
Point to move from (Vertical=No): 13,2,2
Point to move to <default>:  13,2,6
```

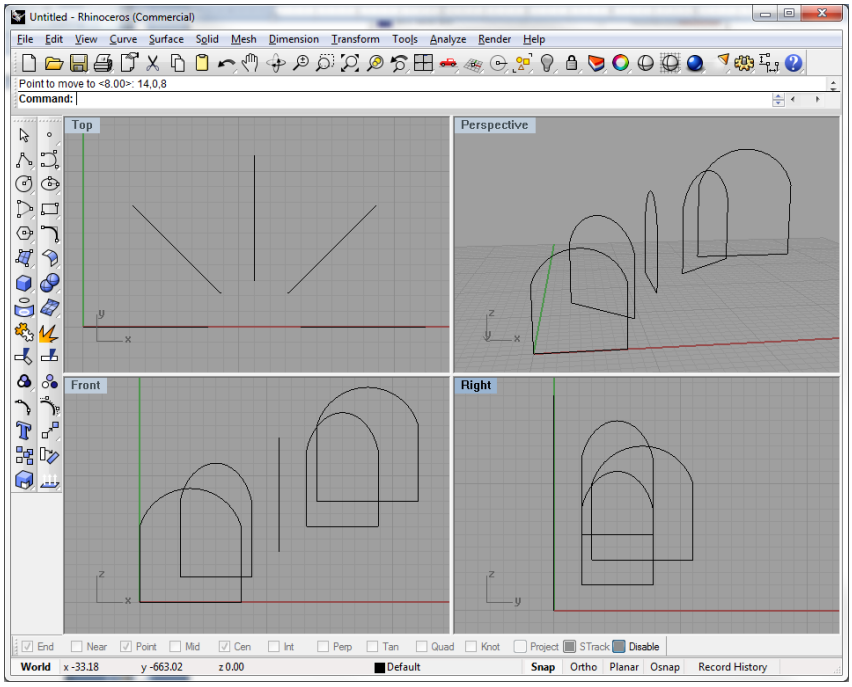
Select the remaining unmoved cross-section.

Select: Transform → Move

Enter the following in the command window.

```
Point to move from (Vertical=No): 14,0,0
Point to move to <default>:  14,0,8
```

With the framework of the tunnel established, it's time to create a solid using the **Loft** command. Before you continue, make sure your structure is identical to the figure below.


Untitled - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Point to move to <8.00>: 14,0,8
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x -33.18 y -663.02 z 0.00 Default Snap Ortho Planar Osnap Record History

Loft

The **Loft** command creates a surface through selected profile curves, in this case the tunnel cross sections.

Select: Surface → Loft

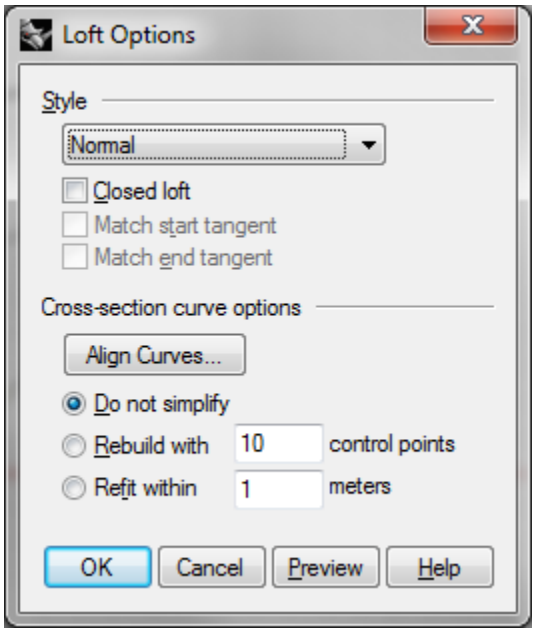
Click and drag a selection window around the five tunnel cross sections and hit Enter. The line and arrows that appear indicate the way in which the surface will be fit through the curves. When using the **Loft** command, it is imperative to ensure that the arrows are facing the same direction and that the curve seams start at the same location on each cross section. It may be convenient to attempt an alternate automatic generation of the curve seams using the **Natural** option.

Enter **Natural** in the command window. The newly generated curve seams are a better representation of our intended tunnel design. Switch between the **Natural** and **Automatic** options to convince yourself that this is the case. Ensure you are using the curve seams created by the **Natural** option and Hit Enter to open the Loft Options

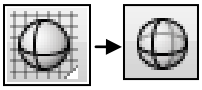
dialog. Note that our five cross sections now have a surface around them!

The Loft Options dialog allows you to customize the style and detail of the fitted surface. For an in-depth description of the various Loft Styles, refer to the *Rhino* Help. The "Normal" style and the "Do not simplify" option in the cross-section curve options are suitable for our tunnel, but feel free to examine the various options using the **Preview** button. When you are ready to continue, ensure "Normal" and "Do not simplify" are selected and hit OK.

✓ *Enter:*
✓ Style: Normal
✓ Do not simplify

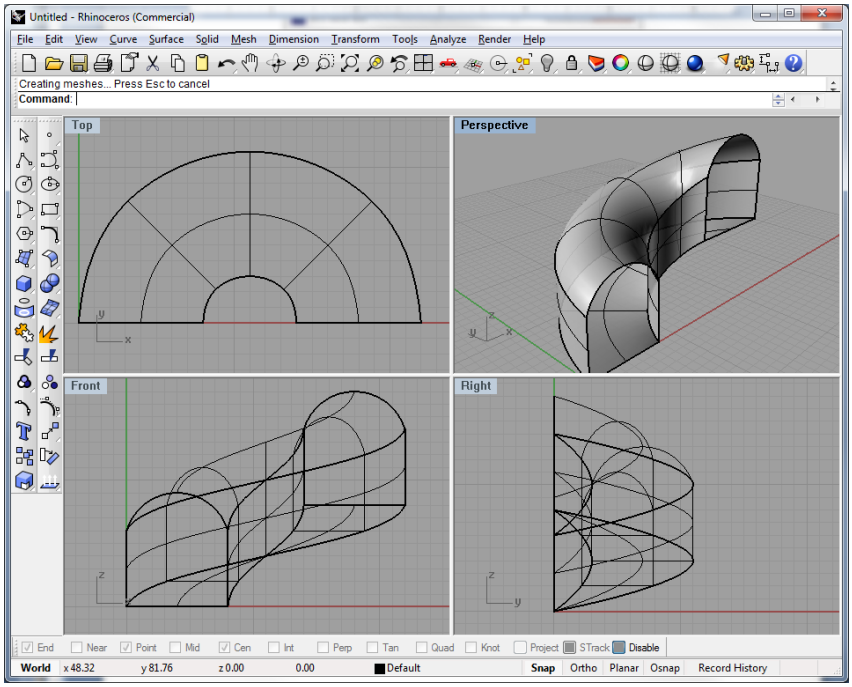


Our tunnel is really coming along! Let's change the view in the "Perspective" view pane to fully appreciate our work. Activate the "Perspective" view pane by clicking anywhere within its bounds.



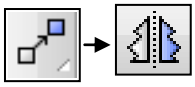
Select: View → Ghosted

The "Ghosted" view makes surfaces translucent, allowing you to see the wireframe as well as the surface of your structure. Take a look at the tunnel in the "Perspective" view by holding down the right mouse button and moving your mouse. You can also move laterally by holding the Shift key and pressing the right mouse button. To reset your view, select View → Set View → Perspective. Your structure should appear as shown below.


Untitled - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Creating meshes... Press Esc to cancel
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x 48.32 y 81.76 z 0.00 0.00 Default Snap Ortho Planar Osnap Record History

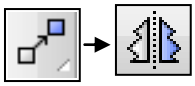
Completing the Spiral Tunnel

Let's create the upper half of our tunnel by mirroring our current structure twice. Select the tunnel.



Select: Transform → Mirror

Ensure that the **Snap** option is selected in the lower portion of the screen. In the "Right" view pane, select the origin of the y-z axis (point 0,0,0) then select a point on the positive z-axis (e.g., 0,0,1). Select the newly created tunnel.

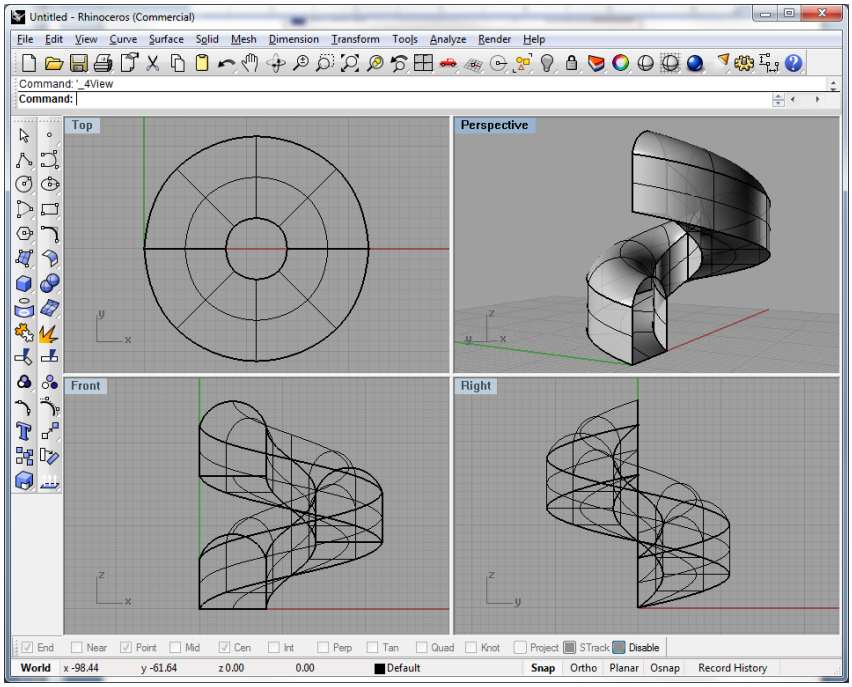


Select: Transform → Mirror

Before selecting a point, turn off the "Copy" option by entering **Copy** in the command window. This way, we are simply rotating our structure as opposed to rotating a copy of it. In the "Top" view pane, select the point at 11,0,0 then select the point at 11,1,0.

We now have the halves of our spiral tunnel! Let's move the newest portion of the tunnel vertically to line up with the older portion.

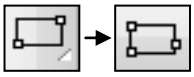
In the "Right" view pane, ensure that the tunnel portion on the left is selected. Click and hold the left mouse button and move the section up (in the z-direction) until the lower mouth of the new portion lines up with the upper mouth of the older portion. Complete the tunnel structure by gluing the two tunnel portions together using the **Join** command (Edit → Join or Ctrl+J). Your structure should appear as shown below.


Untitled - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Command: '_4View
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x -98.44 y -61.64 z 0.00 0.00 Default Snap Ortho Planar Osnap Record History

Creating the Ventilation Shaft and Rectangular Elbow

With the spiral tunnel complete, it's time to build our ventilation shaft. The shaft will be a 20 metre long cylinder connected to the tunnel by a 2 metre long rectangular elbow.

Rectangular Elbow



Select: Curve → Rectangle → 3 Points

Enter the following in the command window.

```
Start of edge (Rounded): 10,10,9
End of edge (Rounded): 12,10,9
Width. Press Enter to use length (Rounded): 12,10,8
```

We can create a three-dimensional shape from this rectangular base using the **ExtrudeCrv** command.

Extrude Curve

Ensure the new rectangular shape is selected.

Select: Surface → Extrude Curve → Straight

The **ExtrudeCrv → Straight** command will extend the curve you select perpendicular to the face it encloses. Entering a negative value will extrude the shape in a direction opposite to the positive axis directions. Set **DeleteInput** in the command window to **YES** then Enter **3**. The "DeleteInput" option deletes the original rectangle, leaving only the three-dimensional extrusion.

Note that although we planned for the elbow to be two metres long, we have extruded it out to three metres in length. This is a necessary consequence of the **Boolean** commands used to join intersecting objects and the fact that the rectangular face of the elbow can not lie flush against the curved tunnel wall.

Boolean Two Objects

Booleans are used to connect two shapes. The difference between **Booleans** and the **Join** command is that the **Join** command simply glues shapes together while **Boolean** commands account for any intersecting space between the connecting shapes.

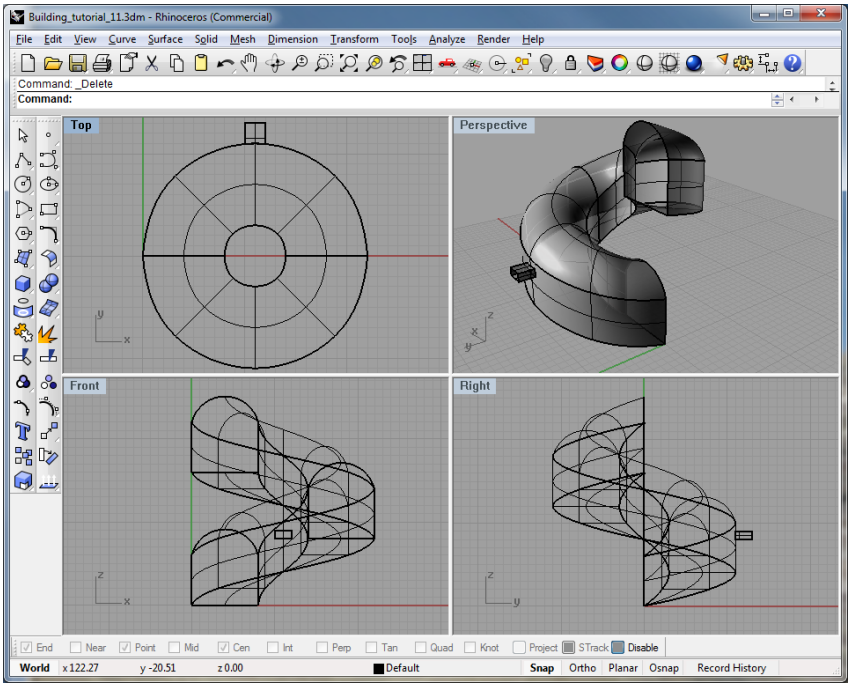
Let's use the **Boolean2Objects** command to attach our rectangular elbow to the tunnel. Let's first select both of our shapes. Select the rectangular elbow then hold the Shift key and select the tunnel.

Select: Solid → Boolean Two Objects

Click through to examine the various Boolean results. Looking at the various intersections, it is clear that the "Union" intersection is what we are looking for. Click through until the command window reads "Union" and hit Enter.

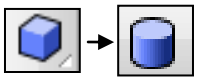
Note that the rectangular elbow is now 2 metres in length! The Boolean intersection has eliminated the shared space between the tunnel and the elbow. We had to use the **Boolean** command here because the **Join** command would not have gotten rid of the section of the tunnel wall connected to the elbow (which would defeat the purpose of our ventilation shaft!).

Your structure should appear as shown below.


Building_tutorial_11.3dm - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Command: _Delete
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x 122.27 y -20.51 z 0.00 Default Snap Ortho Planar Osnap Record History

Ventilation Shaft

With the elbow ready, let's construct the 25 metre cylindrical ventilation shaft. Instead of using the **ExtrudeCrv** command, we'll speed thing up using the **Cylinder** command.



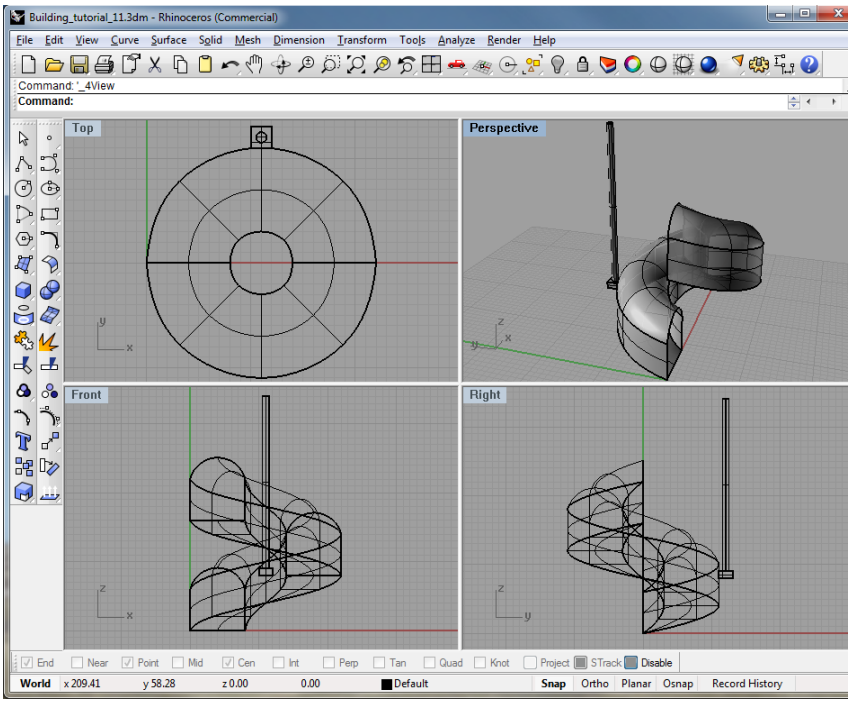
Select: Solid → Cylinder

Enter the following in the command window.

```
Base of cylinder (…): 11,12,9
Radius <default> (Diameter): 0.5
End of cylinder: 25
```

Invoke the **BooleanUnion** command in the command window to join the ventilation shaft and the tunnel structure. In this case, we did not have to artificially extend our cylinder because the circular face lies flush against the rectangular elbow.

Your structure should appear as shown below.


Building_tutorial_11.3dm - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Command: '_4View
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x 209.41 y 58.28 z 0.00 0.00 Default
Snap Ortho Planar Osnap Record History

Cap

To create a mesh acceptable for use in *Examine*^{3D}, it is important that our structure have no open faces. The **Cap** command can be used to close any open faces of a solid. Select the structure.

Select: Solid → Cap Planar Holes

The three open faces of the structure (the two ends of the spiral and the outer face of the rectangular elbow) are now covered.

Mesh

It's time to create our mesh! Ensure the structure is selected.

Select: Mesh → From NURBS object

In the Polygon Mesh Detailed Options Dialog select "Detailed Controls." Set **Density** to 0.5 and **Maximum** and **Minimum edge length** to 0.5, ensuring all other options are zero. Hit Preview to see what the mesh looks like. If we wanted a finer or coarser mesh, we could simply reduce or increase the edge length values. Setting the Maximum and Minimum edge lengths to equal values ensures a mesh will be generated that is effective for use in *Examine*^{3D}. Select OK to generate the mesh.

✓ *Enter:*

✓ Density = 0.5

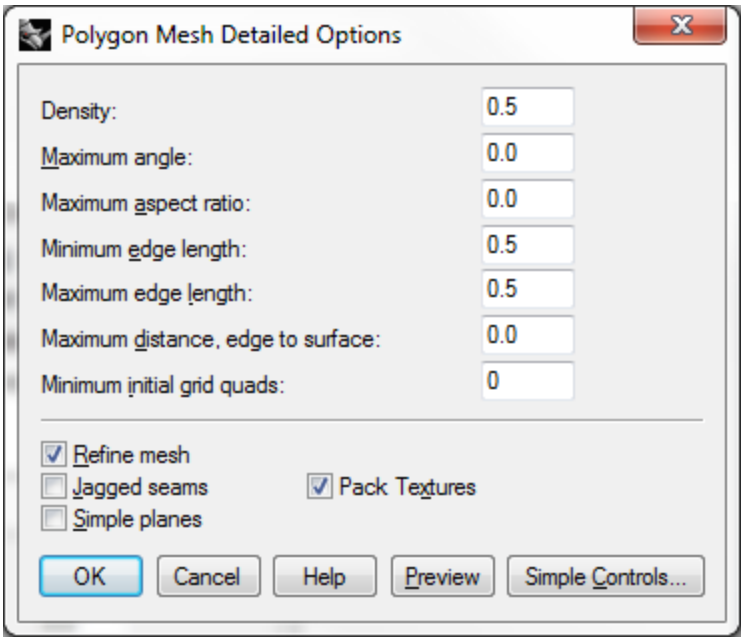
Max. Angle = 0.0

✓ Min. Edge Length = 0.5

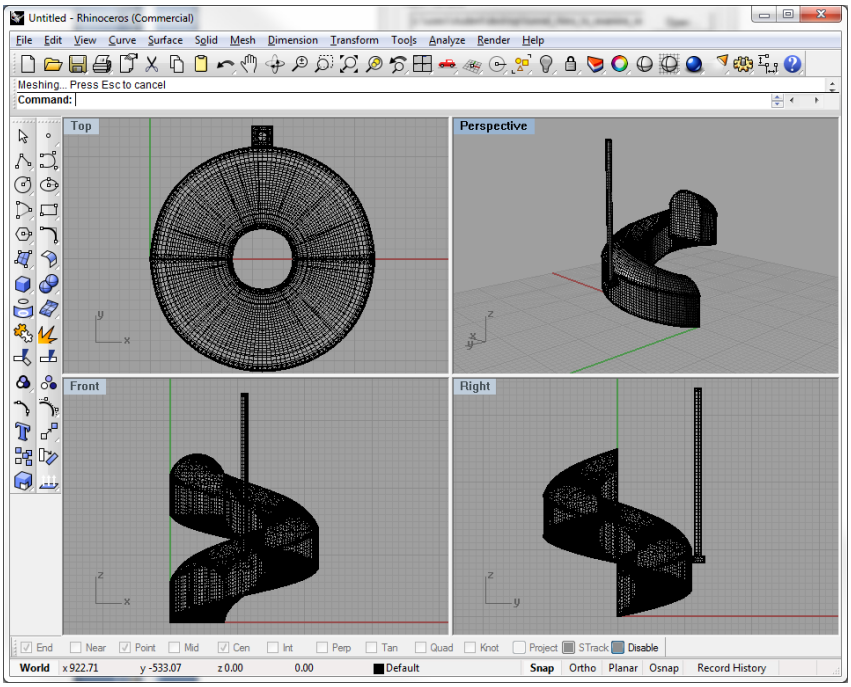
✓ Max. Edge Length = 0.5

Max. Distance = 0.0

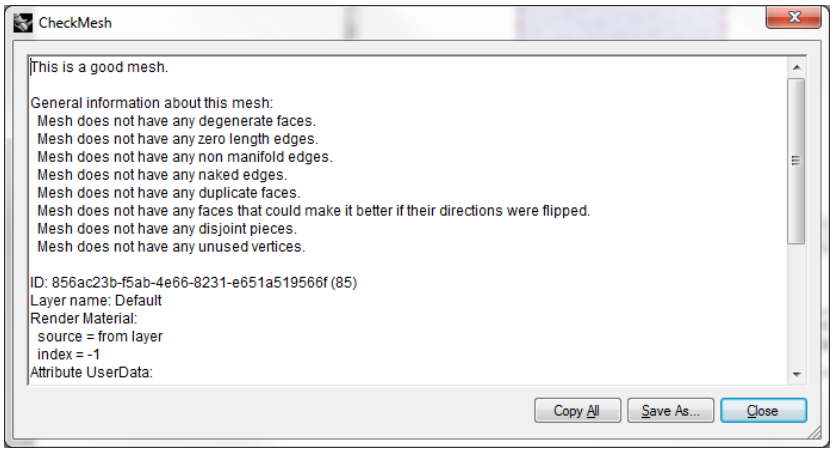
Min. Initial grid quads = 0



Your structure should appear as follows.


Untitled - Rhinoceros (Commercial)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Meshing... Press Esc to cancel
Command:
Top
Perspective
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
World x 922.71 y -533.07 z 0.00 0.00 Default Snap Ortho Planar Osnap Record History

Let's check the quality of the mesh using the **CheckMesh** command. Enter **CheckMesh** in the command window and select the mesh. The CheckMesh dialog should appear as shown below.


CheckMesh
This is a good mesh.
General information about this mesh:
Mesh does not have any degenerate faces.
Mesh does not have any zero length edges.
Mesh does not have any non manifold edges.
Mesh does not have any naked edges.
Mesh does not have any duplicate faces.
Mesh does not have any faces that could make it better if their directions were flipped.
Mesh does not have any disjoint pieces.
Mesh does not have any unused vertices.
ID: 856ac23b-f5ab-4e66-8231-e651a519566f (85)
Layer name: Default
Render Material:
source = from layer
index = -1
Attribute UserData:
Copy All
Save As...
Close

The dialog shows that our mesh is good. Close the dialog when you are ready. For explanations of the mesh errors and repair options, refer to the CheckMesh section in the *Rhino* help window.

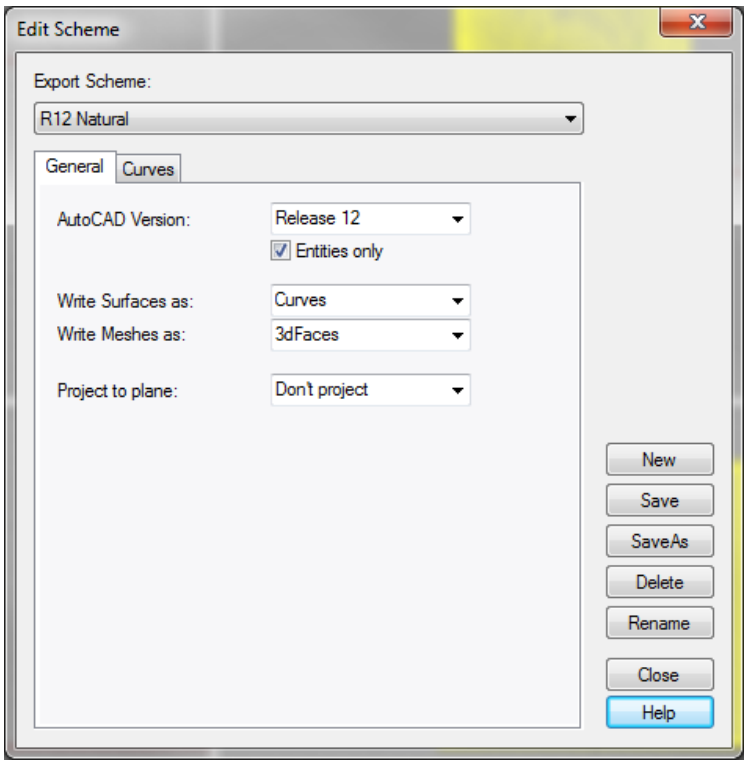
Export structure as .DXF

With our structure meshed correctly, it is time to prepare the file for import into *Examine*^{3D}. The first step in this process is exporting our structure from *Rhino* as a .dxf.

Select: File → Save As

Change the file type to **AutoCad drawing exchange file – (*.dxf)**. Enter 'Tutorial_11' as the file name and select "Save." In the dialog that appears select "Edit Schemes." Select the **R12 Natural** scheme in the menu at the top of the window and make the following changes: set "AutoCad Version" to **Release 12**, toggle the "entities only" option **ON**, and set "Write Meshes as" to **3dFaces**.

- ✓ *Enter:*
- ✓ R12 Natural
- ✓ AutoCad Version: Release 12
- ✓ Entities only
- ✓ Write Meshes: 3dfaces



Save this new export scheme by selecting "SaveAs." Enter "Examine3D" as the scheme name. Select OK and close the "Edit Scheme" dialog. Select "Examine3D" in the "Export AutoCAD File" dialog and hit OK. The structure now exists as a .dxf!

Note that after defining the "Examine3D" export scheme, you need not enter the edit scheme dialog again. Simply select "Examine3D" as your export scheme anytime you are exporting a file for use in *Examine*^{3D}.

Create .GEO File

This new .dxf file can be easily converted to .geo format for import to *Examine*^{3D} using the DXFGEO utility conveniently located in the Examine 3D folder (Start →Rocscience → Examine3D 4.0 → Utilities → DxfGeo).

In the DXFGEO window, open the "Tutorial_11.dxf" file. Toggle "DXF Coordinate Transform" **ON** and change the coordinate as follows: Set Coordinate 1 to "**East**," set Coordinate 2 to "**North**," and set Coordinate 3 to "**Up**." Select Convert to create your .geo file and hit Exit to close the utility.

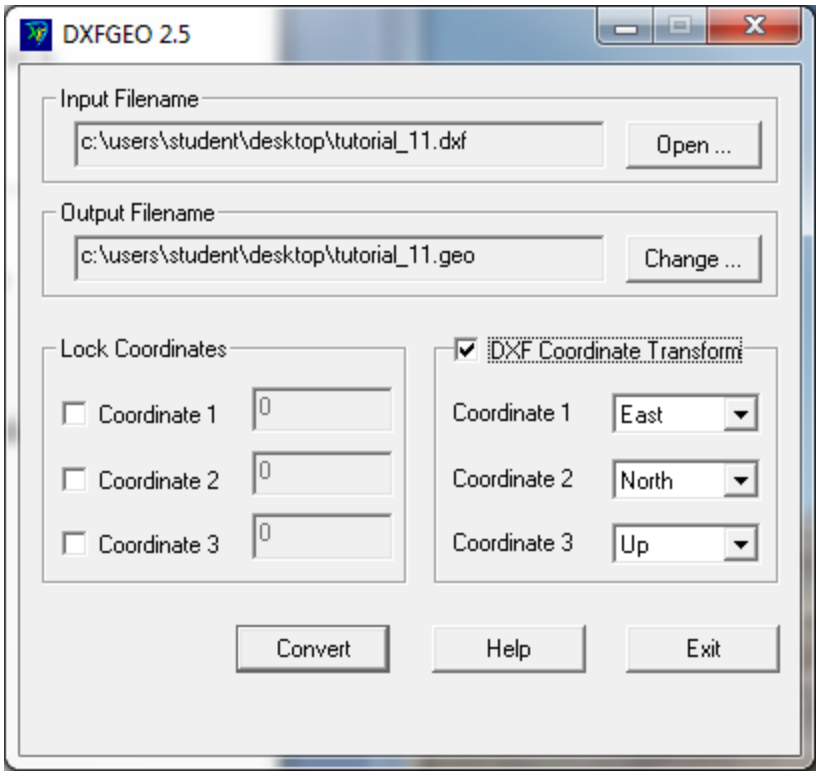
✓ *Enter:*

✓ DXF Coordinate Transform

✓ Coordinate 1 = East

✓ Coordinate 2 = North

✓ Coordinate 3 = Up



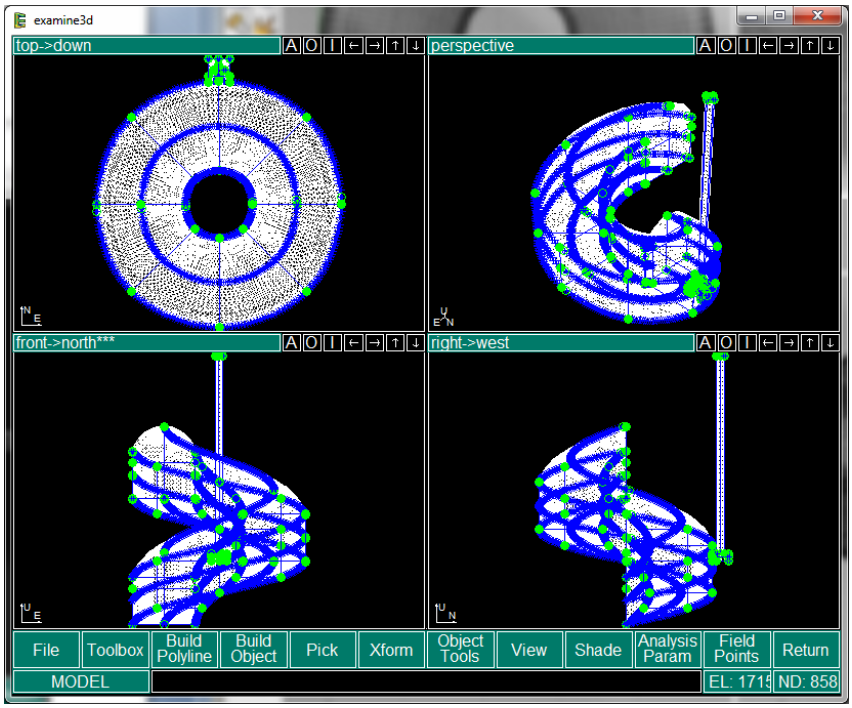
Import Structure into *Examine*^{3D}

The model is now ready for import into *Examine*^{3D}! Start up *Examine*^{3D} by selecting the Examine^{3D} icon in the Start → Rocscience → Examine3D 4.0 menu.

Open the Modeler in Examine^{3D}.

Select File → Append File.

Open the Tutorial_11.geo file we created using the DxfGeo utility and your model will appear!



We're not done just yet however; the mesh must pass the mesh quality check in *Examine*^{3D} to be valid for analysis.

Select Toolbox → Object Check.

Hit Enter to proceed through the check. When the check displays the "Geometry is NOT LEAKY" notification, the check is complete and the structure is ready for analysis!

Note that LEAKY geometry is the biggest concern in the mesh check. If you encounter this error you must return to *Rhino* and repair your model.

In *Rhino*, use the **CheckMesh** command and try to resolve any listed errors (especially Naked Edges) using the methods provided in the CheckMesh section of the *Rhino* help library. Of particular note are the **ShowEdges** and **FillMeshHole** commands which can be used to find and plug any holes in your geometry.

This concludes the "Importing a model from Rhinoceros 4.0" tutorial.